NI Multisim™

Fundamentals

January 2012
372937C-01

Worldwide Technical Support and Product Information

`ni.com`

Worldwide Offices

Visit `ni.com/niglobal` to access the branch office Web sites, which provide up-to-date contact information, support phone numbers, email addresses, and current events.

National Instruments Corporate Headquarters

11500 North Mopac Expressway   Austin, Texas 78759-3504   USA   Tel: 512 683 0100

For further support information, refer to the *Technical Support and Professional Services* appendix. To comment on National Instruments documentation, refer to the National Instruments Web site at `ni.com/info` and enter the Info Code `feedback`.

Important Information

Warranty

Copyright

Trademarks

Patents

For patents covering National Instruments products/technology, refer to the appropriate location: **Help»Patents** in your software, the `patents.txt` file on your media, or the *National Instruments Patent Notice* at `ni.com/patents`.

Some portions of this product are protected under United States Patent No. 6,560,572.

Export Compliance Information

Refer to the *Export Compliance Information* at `ni.com/legal/export-compliance` for the National Instruments global trade compliance policy and how to obtain relevant HTS codes, ECCNs, and other import/export data.

WARNING REGARDING USE OF NATIONAL INSTRUMENTS PRODUCTS

(1) NATIONAL INSTRUMENTS PRODUCTS ARE NOT DESIGNED WITH COMPONENTS AND TESTING FOR A LEVEL OF RELIABILITY SUITABLE FOR USE IN OR IN CONNECTION WITH SURGICAL IMPLANTS OR AS CRITICAL COMPONENTS IN ANY LIFE SUPPORT SYSTEMS WHOSE FAILURE TO PERFORM CAN REASONABLY BE EXPECTED TO CAUSE SIGNIFICANT INJURY TO A HUMAN.

(2) IN ANY APPLICATION, INCLUDING THE ABOVE, RELIABILITY OF OPERATION OF THE SOFTWARE PRODUCTS CAN BE IMPAIRED BY ADVERSE FACTORS, INCLUDING BUT NOT LIMITED TO FLUCTUATIONS IN ELECTRICAL POWER SUPPLY, COMPUTER HARDWARE MALFUNCTIONS, COMPUTER OPERATING SYSTEM SOFTWARE FITNESS, FITNESS OF COMPILERS AND DEVELOPMENT SOFTWARE USED TO DEVELOP AN APPLICATION, INSTALLATION ERRORS, SOFTWARE AND HARDWARE COMPATIBILITY PROBLEMS, MALFUNCTIONS OR FAILURES OF ELECTRONIC MONITORING OR CONTROL DEVICES, TRANSIENT FAILURES OF ELECTRONIC SYSTEMS (HARDWARE AND/OR SOFTWARE), UNANTICIPATED USES OR MISUSES, OR ERRORS ON THE PART OF THE USER OR APPLICATIONS DESIGNER (ADVERSE FACTORS SUCH AS THESE ARE HEREAFTER COLLECTIVELY TERMED “SYSTEM FAILURES”). ANY APPLICATION WHERE A SYSTEM FAILURE WOULD CREATE A RISK OF HARM TO PROPERTY OR PERSONS (INCLUDING THE RISK OF BODILY INJURY AND DEATH) SHOULD NOT BE RELIANT SOLELY UPON ONE FORM OF ELECTRONIC SYSTEM DUE TO THE RISK OF SYSTEM FAILURE. TO AVOID DAMAGE, INJURY, OR DEATH, THE USER OR APPLICATION DESIGNER MUST TAKE REASONABLY PRUDENT STEPS TO PROTECT AGAINST SYSTEM FAILURES, INCLUDING BUT NOT LIMITED TO BACK-UP OR SHUT DOWN MECHANISMS. BECAUSE EACH END-USER SYSTEM IS CUSTOMIZED AND DIFFERS FROM NATIONAL INSTRUMENTS' TESTING PLATFORMS AND BECAUSE A USER OR APPLICATION DESIGNER MAY USE NATIONAL INSTRUMENTS PRODUCTS IN COMBINATION WITH OTHER PRODUCTS IN A MANNER NOT EVALUATED OR CONTEMPLATED BY NATIONAL INSTRUMENTS, THE USER OR APPLICATION DESIGNER IS ULTIMATELY RESPONSIBLE FOR VERIFYING AND VALIDATING THE SUITABILITY OF NATIONAL INSTRUMENTS PRODUCTS WHENEVER NATIONAL INSTRUMENTS PRODUCTS ARE INCORPORATED IN A SYSTEM OR APPLICATION, INCLUDING, WITHOUT LIMITATION, THE APPROPRIATE DESIGN, PROCESS AND SAFETY LEVEL OF SUCH SYSTEM OR APPLICATION.

Conventions

The following conventions are used in this manual:

»

The » symbol leads you through nested menu items and dialog box options to a final action. The sequence **Tools»Clear ERC Markers»Entire design** directs you to pull down the **Tools** menu, select the **Clear ERC Markers** item, and select **Entire design** from the resulting dialog box.



This icon denotes a note, which alerts you to important information.

bold

Bold text denotes items that you must select or click in the software, such as menu items and dialog box options. Bold text also denotes parameter names.

italic

Italic text denotes variables, emphasis, a cross-reference, or an introduction to a key concept. Italic text also denotes text that is a placeholder for a word or value that you must supply.

`monospace`

Text in this font denotes text or characters that you should enter from the keyboard, sections of code, programming examples, and syntax examples. This font is also used for the proper names of disk drives, paths, directories, programs, subprograms, subroutines, device names, functions, operations, variables, filenames, and extensions.

Contents

Chapter 6
Data Analysis

Chapter 7
PCB Layout

Chapter 8
Automation API

Chapter 9
Multisim MCU

Appendix A
Technical Support and Professional Services

Index

1

Introduction

National Instruments Circuit Design Suite is a suite of EDA (Electronics Design Automation) tools that assists you in carrying out the major steps in the circuit design flow.

Multisim is the schematic capture and simulation program designed for schematic entry, simulation, and feeding to downstage steps, such as PCB layout. It also includes mixed analog/digital simulation capability, and microcontroller co-simulation.

This manual describes some of the basic features of Multisim.

Some of the described features may not be available in your edition of Multisim. Refer to the *NI Circuit Design Suite Release Notes* for a description of the features available in your edition.

Refer to the *Multisim Help* for complete information on the features described in this manual.

2

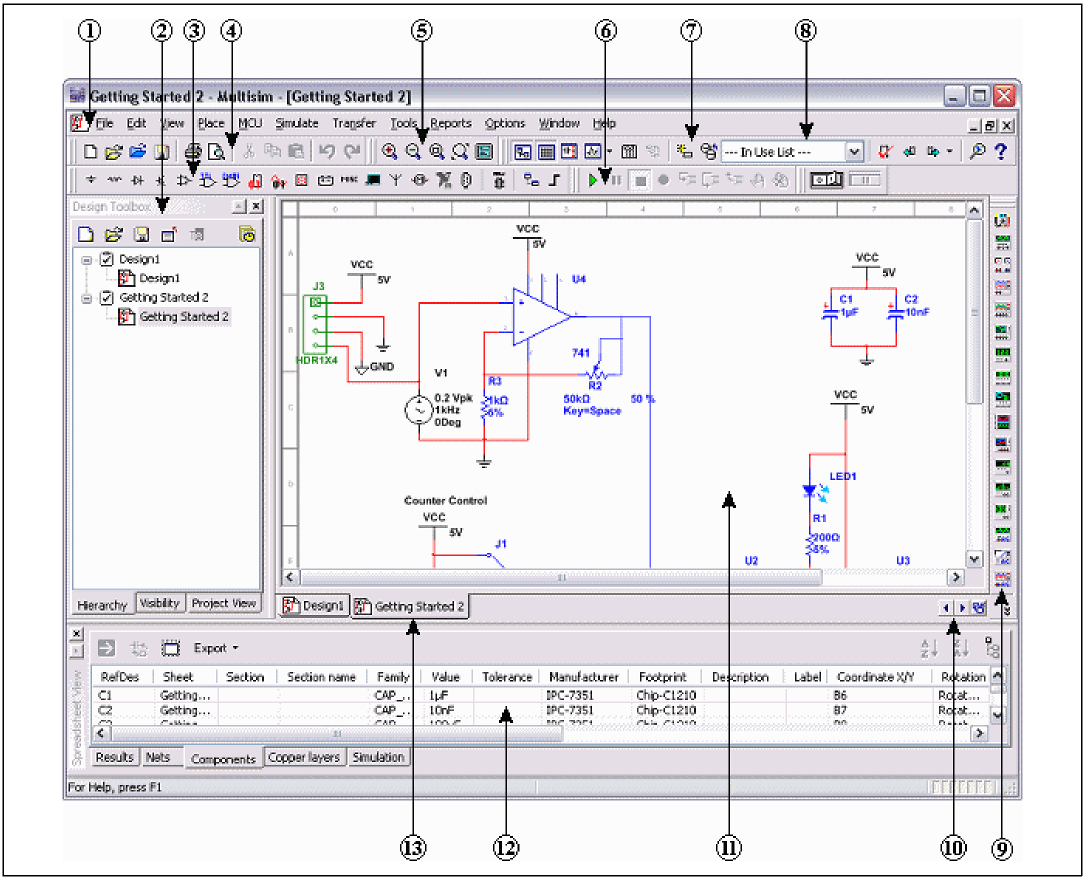
User Interface

This chapter introduces you to Multisim's user interface.

Some of the described features may not be available in your edition of Multisim. Refer to the *NI Circuit Design Suite Release Notes* for a description of the features available in your edition.

Refer to the *Multisim Help* for complete information on the features described in this chapter.

Introduction



1	Menu Bar	5	View Toolbar	8	In Use List	11	Circuit Window
2	Design Toolbox	6	Simulation Toolbar	9	Instruments Toolbar	12	Spreadsheet View
3	Component Toolbar	7	Main Toolbar	10	Scroll Left/Right	13	Active Tab
4	Standard Toolbar						

Multisim is the schematic capture and simulation application of National Instruments Circuit Design Suite, a suite of EDA (Electronics Design Automation) tools that assists you in carrying out the major steps in the circuit design flow. Multisim is designed for schematic entry, simulation, and feeding to downstage steps, such as PCB layout.

Note For detailed information about the Multisim user interface, and information on the other components of NI Circuit Design Suite, refer to the *Multisim Help* and the *Getting Started with NI Circuit Design Suite* manual.

Toolbars

Multisim includes the following toolbars:

- **Standard Toolbar**—Contains buttons for commonly-performed Windows functions.
- **Main Toolbar**—Contains buttons for commonly-performed Multisim functions.
- **Simulation Toolbar**—Contains buttons used during simulation.
- **View Toolbar**—Contains buttons used to manipulate the appearance of the workspace.
- **Components Toolbar**—Contains buttons used to place components onto the workspace.
- **Virtual Toolbar**—Contains buttons used to place virtual components on the workspace.
- **Graphic Annotation Toolbar**—Contains buttons used to draw graphic elements such as lines and circles on the workspace. You can also place comments and pictures from here.
- **Instruments Toolbar**—Contains buttons used to place instruments on the workspace.

Setting Schematic Capture Preferences

You can customize virtually any aspect of Multisim preferences, including the colors used in your circuit, page size, zoom level, auto-backup interval, symbol set (ANSI or DIN) and printer setup. Your customization settings are saved individually with each circuit file you use so you could, for example, have one color scheme for one circuit and another for a different circuit.

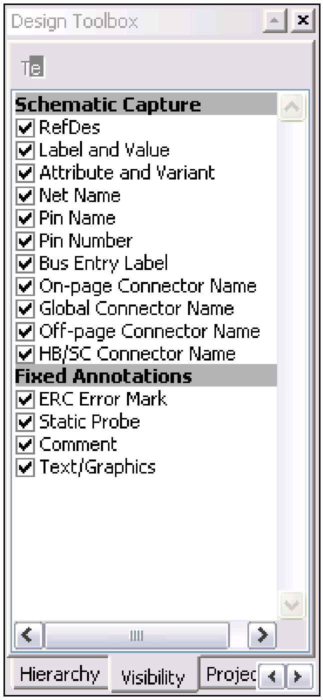
Customization is done through the:

- **Global Preferences** dialog box—Use to set up global preferences. These preferences can vary from computer to computer.
- **Sheet Properties** dialog box—Use to set up the preferences for the active sheet. These preferences are saved with the circuit files so that if the circuit is opened on another computer, it will use the same settings.

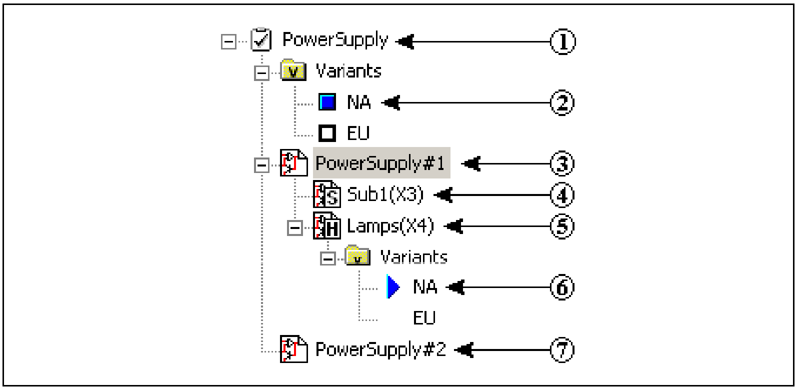
Design Toolbox

Use the **Design Toolbox** to manage various elements in the schematic.

You can choose which layers to display on the current worksheet from the **Visibility** tab, as shown in the figure below.



The **Hierarchy** tab contains a tree that shows the files in the design that you have open, as shown in the figure below.



1	Design Root	5	Hierarchical Blocks
2	Active Variant	6	Included in Active Variant
3	Page 1 of Multi-sheet	7	Page 2 of Multi-sheet
4	Subcircuit		

The **Project View** tab is where you can see the files assigned to a project. A project is a collection of files. For example, all the circuit files making up a design may be grouped together in a project, as well as external design documentation, simulation output, reports, and PCB layouts generated with Ultiboard.

For detailed information on the above, refer to the *Multisim Help*.

Customizing the Interface

The Multisim user interface is highly customizable. Separate customizations can be applied whenever a different type of sheet becomes active. For example, the toolbars and docking windows can be re-configured as you move from a circuit sheet to a description sheet.

Toolbars can be docked in various positions and orientations. The contents of the toolbars can be customized. New toolbars can be created. The menu system is fully customizable, including all context menus for the various object types.

The keyboard shortcut system is also customizable. This allows for any keys or key combinations to be assigned to any command that can be placed in a menu or on a toolbar.

3 Schematic Capture

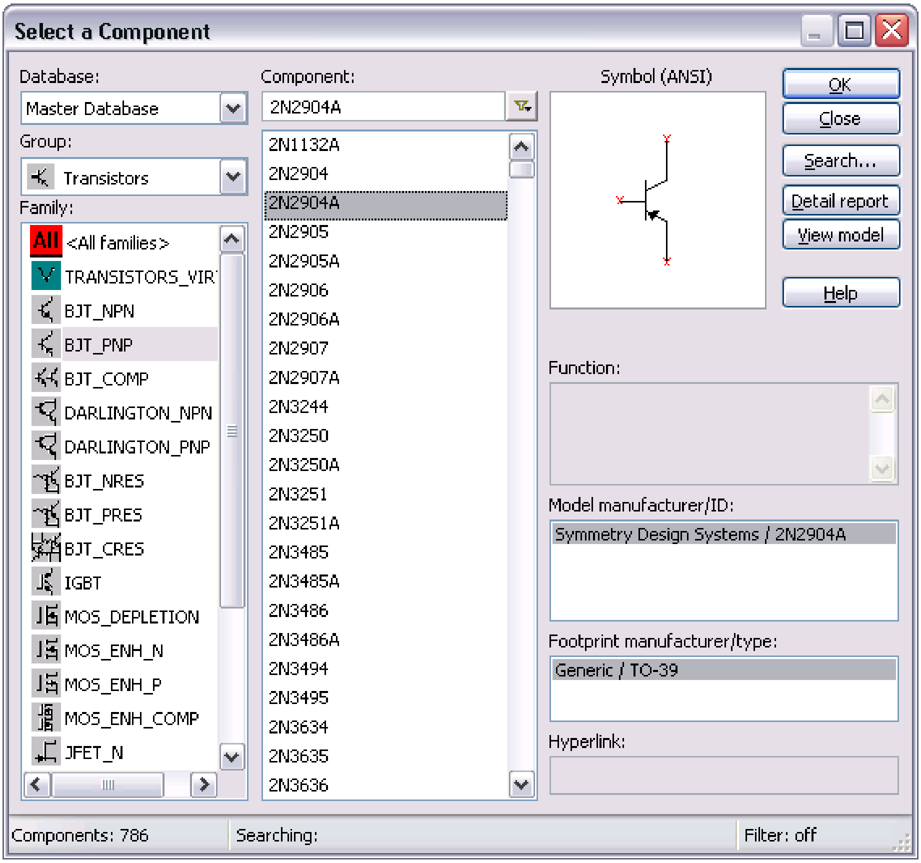
Schematic capture is the first stage in developing your circuit.

This is where you choose the components you want to use, place them on the circuit window in the desired position and orientation, wire them together, and otherwise prepare your design. You can modify component properties, orient your circuit on a grid, add text and a title block, add subcircuits and buses, and control the color of the circuit window background, components and wires.

Refer to the *Multisim Help* for complete information about schematic capture.

Placing Components

The first step in schematic capture is to place the appropriate components on your circuit window. You typically do this via the component browser (that is, the **Select a Component** dialog box). Refer to the figure below.



Each button on the **Components** toolbar corresponds to a group of components with similar functionality. Click one of these buttons to open the component browser and select the desired component.

You can also select **Place»Component** to display the **Select a Component** dialog box. Components are organized by database, group, and family (for example, **Master Database**, **Sources Group**, **Power Sources Family**). Use type-ahead to type a few characters to jump to the component you are looking for. Search capabilities allow you to find components using generalized wildcard searches throughout all the databases.

Each time you place a component, it is added to the **In Use List**. To place another instance of any currently placed component, select it from the list and click to place it in the desired location.

After you have placed a component, you can rotate or flip it to the desired orientation.

Wiring Components

To create a basic wire, click on any one of a placed component's symbol pins and click on a pin on another component to complete the connection.

You can also double-click anywhere in a circuit to start a wire. This creates a junction at that location and starts the wiring from that point.

You have the option to make an automatic connection if a component pin is touching a wire, another pin or a junction.

Each time a wire is placed it either creates a new *net*, or joins an existing one. A *net* is a collection of wires all of which define a common electrical potential. The term net is typically used to describe this concept when discussing PCB layout. This is equivalent to the concept of a *node* when discussing SPICE-based circuit simulation.

After you have placed a wire, you can change its path by clicking on it and dragging any of the selection handles that appear. Wires can easily be disconnected from one location and moved to another. You may also change the color of part or all of a wire.

Labeling

There are many ways that you can label your designs in Multisim:

- **Adding a Title Block**—You can use the **Title Block Editor** to create customized title blocks. If desired, a title block can be included on every page of your design.

  Various fields in the title block are automatically completed depending on the context and various document properties. When designing the title block, you can choose a pre-defined field or create your own. You choose appropriate fonts depending upon your language of preference.

  Title blocks can include elements such as text, lines, arcs, bezier curves, rectangles, ovals, arcs, and bitmaps.

- **Adding Miscellaneous Text**—You can use the **Place»Text** command to place text anywhere on the workspace.
- **Adding a Comment**—You can add comments to show engineering change orders, to facilitate collaborative work among team members, or to allow background information to be attached to a design.

  You can "pin" a comment to the workspace, or directly to a component. When a component with a "pinned" comment is moved, the comment also moves.
- **Using Graphic Annotation**—Use this functionality to add the following graphic elements to your workspace:
  - Text
  - Line
  - Multiline
  - Rectangle
  - Ellipse
  - Arc
  - Polygon
  - Picture
  - Comment
- **Modifying Net Names**—Multisim automatically assigns a net name to each node in the circuit. If desired, you can modify a net name to something more meaningful to the circuit design. For example, you may wish to change a net name to "Output."

  If you are planning to modify net names for certain "global" reserved nets such as VCC and GND, there are some cautions that apply. Refer to the *Multisim Help* for more information.
- **Using the Description Box**—In addition to adding text to a particular portion of a circuit, you can add general descriptions to your circuit using the **Description Box**. You can also place bitmaps, sound and video in the **Description Box**.

  Along with descriptive text and graphics, the **Description Box** lets you synchronize scrolling and display of text, graphics or video with events in the simulation. You do this by placing measurement probes in your design and setting the probe properties with the events that you would like to have synchronized with the scrolling or display of text, graphics or video.

- **Linking a Form to a Circuit**—You can use Multisim's form functionality to send circuits for approval, design reviews, or anywhere feedback on a design is needed. Once the form has been completed, the circuit file, including the completed form, can be returned to the originator via email.

  A form can include any of the following types of questions:

  – **Multiple Choice**—The correct response is selected from two or more possible responses.

  – **True/False**—A statement is made on the form and either **True** or **False** is selected as the response.

  – **Data Entry**—A field for limited data entry is provided.

  – **Free Form**—A field for more extensive data entry is provided.

  You can also insert links in the **Description Box** that, when double-clicked, take the person completing the form directly to a linked question. This is useful where clarification or further explanation of a specific question may require more text than you want to put on the form.

Placed Component Properties

Each placed component has a set of properties that control certain aspects of it beyond those stored in the Multisim database. These properties affect only the placed component, not other instances of that component in other circuits or other locations in this circuit.

To modify these properties, double-click on the placed component to display its properties dialog box, and select one of the following tabs:

- **Label**—This is where you change the selected component's **RefDes**, **Label**, and **Attributes**. An attribute can be any name or value you choose. For example, you could use the manufacturer name or a name that is meaningful to you such as "new resistor" or "revised May 15."
- **Display**—This is where you change what elements of the selected component display on the workspace, for example, **RefDes**, **Value**.
- **Value**—This shows the value/model being used for the placed component. You can also change the footprint used for most components from here. The contents of this tab differ depending on the selected component. Refer to the *Multisim Help* for detailed information.

For some components, such as power sources, you can set how they are used in any analyses you might perform on the circuit. These components offer additional parameters in the **Value** tab.

- **Fault**—You may want to assign faults to analog components for instructional purposes, such as troubleshooting exercises. Use this tab to assign faults to specific pins of the selected component. You can open or short-circuit a pin, or assign leakage—a resistance in parallel with selected terminals which causes current to leak past the terminals instead of going through them.

  You cannot assign faults to digital components.
- **Pins**—Before running an **Electrical Rules Check** (**ERC**) you can set which pins to exclude or include for the selected component in this tab. You can add **NC** (no connection) markers to a pin. A pin marked in this way will not have an unconnected pin error assigned to it when you run the **ERC**. Also, you cannot wire to pins with NC markers.
- **Variant**—A variant is a specific version of a circuit. For example, you may have a design with botha North American, and a European power supply. Each of these would be one variant of the circuit.
- **User Fields**—Components have up to 20 user fields that provide user-specific information about the component (for example, vendor, manufacturer, hyperlink). You can change the **Value** of a user field from this tab. The titles of the user fields are entered and edited from the **Database Manager**. Refer to the *Multisim Help* for information about the **Database Manager**.

Spreadsheet View

Use the **Spreadsheet View** for fast viewing and editing of parameters including component details such as footprints, reference designators, attributes and design constraints.

The **Spreadsheet View** contains the following tabs:

- **Results**—The results of most operations appear here, for example, Electrical Rules Checks, importing circuits.
- **Nets**—This is where you can find and edit information about the nets in your design.
- **Components**—This is where you can find and edit information about the components in your design.

- **Copper layers**—This is where you can find and edit trace routing information for when your design is transferred to Ultiboard, Circuit Design Suites's PCB layout program.
- **Simulation**—This is where errors and warnings from SPICE netlist checks, as well as simulation errors appear.

Note Some of the above-mentioned tabs may not be available in your edition of Multisim. Refer to the *NI Circuit Design Suite Release Notes* for a list of the features in your edition.

Electrical Rules Checking

Once you have wired your circuit, you can check the connections for correctness based on the rules set up in the **Electrical Rules Check** dialog box. **Electrical Rules Checking** creates and displays a report detailing connection errors (such as an output pin connected to a power pin) and unconnected pins.

Depending on your circuit, you may wish to have warnings issued if some types of connections are present, error messages for other connection types, and no warnings or errors for other connections. You control the type of connections that are reported by setting up the rules in the grid found in the **ERC Rules** tab of the **Electrical Rules Check** dialog box.

When you run an electrical rules check (ERC), errors and warnings display in a results pane at the bottom of the screen and the circuit is annotated with circular error markers. Double-clicking on an error in the results pane centers and zooms on the error's location in the circuit.

Working with Larger Designs

Multisim includes many features that simplify working with larger designs:

- **Flat Multi-Sheet Design**—In many instances circuit designs are too large to fit all components on a single sheet or for logical reasons it's easier to think of a circuit design if it is divided. In this case, you can use this feature to place off-page connectors between different sections of your circuit.
- **Hierarchical Design**—You can use hierarchical blocks and subcircuits to organize functionally related pieces of a design into manageable pieces. Multisim's hierarchical functionality allows you to build a hierarchy of inter-connected circuits, increasing the reusability of your circuit designs and ensuring consistency across a group of designers. For example, you might build a library of commonly used

circuits, stored in a central location. Those circuits could in turn be contained in other, more complex circuits, which could be used to create yet another level of circuit design.

Hierarchical blocks and subcircuits are similar except that subcircuits are saved with the original circuit and hierarchical blocks are individual circuit files that are referenced from a main file.

- **Buses**—To simplify wiring, use buses to carry multiple nets within a page, across pages, and down into subcircuits and hierarchical blocks.

  Use **Bus Vector Connect** to connect multiple pins of a chip to a bus in a single operation. This is the preferred method for placing numerous connections from a multi-pinned device, such as an IC, to a bus.
- **Variants**—A variant is a specific version of a circuit. As PCBs are manufactured for distribution on a global scale, some designs may require modifications depending on their target markets. For example, power supply requirements for the European market differ from those of North America. The variations in the power supply requirements may call for the use of different components in a design. You may wish to produce a single PCB which would meet requirements for both the North American and European versions. The board itself must contain the traces as well as land patterns/footprints for both variations of the design. The PCB would then be populated with components according to the target market of the device.
- **Project Management**—To help manage the various files associated with a design, you can use the project management functionality. A project is a collection of files. For example, all the circuit files making up a design may be grouped together in a project, as well as external design documentation, simulation output, reports, and PCB layouts generated with Ultiboard.

Circuit Wizards

Use Multisim's circuit wizards to generate circuits that contain a schematic diagram, simulation models and a netlist. Once you have built the circuit, it can be simulated in the usual manner.

The wizards that are available are:

- **555 Timer Wizard**—Use to build astable and monostable oscillator circuits that use the 555 timer. Astable operation produces a free-running oscillator that does not need any input signal. Monostable operation produces a single output pulse in response to an input trigger

pulse. When an input signal is applied, each input pulse produces one output pulse.

- **Filter Wizard**—Use to design numerous types of filters, including passive and active **Butterworth** and **Chebyshev** filters.
- **Common Emitter BJT Amplifier Wizard**—Use to design common emitter amplifier circuits.
- **Op-amp Wizard**—Use to design the following op-amp circuits:
  - Inverting Amplifier
  - Non-inverting Amplifier
  - Difference Amplifier
  - Inverted Summing Amplifier
  - Non-inverted Summing Amplifier
  - Scaling Adder

4 Components and Database

This chapter introduces you to the underlying structure and organization of the Multisim component database. It also discusses the **Database Manager**, and **Component Editing**, including **Model Makers** and **Code Modeling**.

Some of the features described in this chapter may not be available in your edition of Multisim. Refer to the *NI Circuit Design Suite Release Notes* for a list of the features in your edition.

Refer to the *Multisim Help* for complete information on the features described in this chapter.

About the Database

The Multisim component database holds the information necessary to describe any component. It contains everything needed for schematic capture (symbols), simulation (models) and PCB layout (footprints), as well as other electrical information.

There are three levels of database provided by Multisim. The **Master Database** is read-only, and contains components supplied by National Instruments. The **User Database** is private to an individual user. It stores components built by an individual that are not intended to be shared. The **Corporate Database** stores custom components that are intended to be shared across an organization. Various database management tools are available to move components between databases, merge databases, and edit them.

All the components in the databases are divided into groups and then into families within those groups.

When you choose a component from the database and drop it onto the circuit, a copy of the component is placed onto the circuit. Any edits made to the component in the circuit do not affect the original database copy, or any similar components previously placed on the circuit. Likewise, any edits made to the component in the database after a copy has been placed

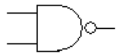
do not affect the previously placed components, but will affect all subsequently placed ones.

Multisim divides components into logical groups. Each **Group** contains families of related components. The groups are:

- Sources
- Basic
- Diodes
- Transistors
- Analog
- TTL
- CMOS
- MCU
- Advanced Peripherals
- Misc Digital
- Mixed
- Indicators
- Power
- Misc
- RF
- Electro-mechanical
- Connectors
- NI Components

Each component has pre-defined information that is visible when you invoke the **Select a Component** dialog box during schematic capture:

Field	Description	Example
Database	Name of Multisim database in which the component is stored.	User
Group	Name of the group to which the component belongs.	TTL
Family	Name of family to which the component belongs.	74S
Component	Name of the individual component.	74S00D

Field	Description	Example
Symbol	Symbol used (either ANSI or DIN) to represent the component during schematic capture.	
Function	Describes the component. Not available for resistors, inductors or capacitors.	QUAD 2-INPUT NAND
Component type	Describes the construction of resistors, inductors or capacitors only. Contents vary depending on component selection. Does not display for all components.	Carbon film (for a resistor).
Tolerance	Percent tolerance for the resistor, inductor or capacitor selected in the component list. Does not display for other component types.	0.5
Model manufacturer/ID	Name of the company that manufactures the component and the component's ID.	Texas Instruments/74S00
Footprint manufacturer/ Type	Footprint for the component and the package type. Used in Ultiboard or other vendors' PCB layout products.	IPC-7351/DO14
Hyperlink	Link to the vendor's website.	`www.analog.com`

Refer to the *Multisim Help* for detailed information about the Multisim databases, and schematic capture.

Database Manager

Multisim's database is managed through the **Database Manager** dialog box, where you can:

- add and remove component families from the **User Database** or **Corporate Database**.

Note You cannot add or remove families in the **Master Database**.

- set up and modify user fields and their titles for any database.

- add and change component toolbar button symbols for component families in the **User Database** or **Corporate Database**.

Refer to the *Multisim Help* for detailed information about the **Database Manager**.

Component Editing

You can edit components in Multisim using the:

- **Component Wizard**—Use to create and edit new components. This tool quickly steps you through the process of creating a component for use in schematic capture, as well as simulation, layout, or both.
- **Component Properties** dialog box—Use to edit existing components and is accessed from the **Database Manager** dialog box. You can also double-click on a placed component and click **Edit component in DB** to display the **Component Properties** dialog box.

  The **Component Properties** dialog box contains the following tabs:

  – **General** tab—General information such as name, manufacturer, date and author.

  – **Symbol** tab—Use this tab to edit a component's symbol, give a component the same symbol as another component or create a new symbol for a component. You can also access the **Symbol Editor** from here. This is a specialized graphics editor that you can use to create or modify a component's symbol.

  – **Model** tab—Model information is necessary only for a component that will be simulated. Refer to the *Creating a Component Model Using the Model Makers* and *Creating a Model Using Code Modeling* sections for more information.

  – **Pin parameters** tab—This is is where you assign the pin type for each of the symbol pins in a component. The **Symbo pin** is the name of the pin. For example, most digital components have pins named "Vcc" and "GND." You also select whether or not you wish to include each symbol pin in the **Electrical Rules Check**.

  – **Footprint** tab—The footprint is the package that Multisim uses when exporting a schematic containing this component to a PCB Layout package such as Ultiboard. This tab is where you can specify/modify package information for a component and map footprint and symbol pins of a component.

  – **Electronic parameters** tab—This tab consists of two sets of fields. The **Device Specific Parameters** vary depending on the

type of component. The **Common Parameters** are common to all components. The parameters displayed in this tab do not affect simulation results. There are only used for detail reports.

– **User fields** tab—Use this tab to edit **User Field** data.

Creating a Component Model Using the Model Makers

Multisim offers advanced **Model Makers** which automatically generate simulation models when you give them databook values as input. **Model Makers** will save you time and effort but do require experience for you to become proficient with them. You do not need to be familiar with SPICE to use the **Model Makers**.

For each **Model Maker**, preset values are provided for a specific model. However, you can select numerical values based on the component you are interested in, using data from a databook.

The following **Model Makers** are available:

- AC Motor
- BJT
- Boost Converter
- Buck Boost Converter
- Buck Converter
- Cuk Converter
- Diode
- Ideal Transformer (Multiple Winding)
- Linear Transformer (Multiple Winding)
- Linear Transformer with Neutral Terminal
- Two Winding Linear Transformer
- Non-linear Transformer (Multiple Winding)
- MOSFET
- Operational Amplifier
- Silicon Controlled Rectifier (SCR)
- Zener Diode
- Waveguide
- Microstrip Line
- Open End Microstrip Line

- RF Spiral Inductor
- Strip Line Model
- Stripline Bend
- Lossy Line
- Interdigital Capacitor

Creating a Model Using Code Modeling

The behavior of some components is difficult to model using SPICE components. In these cases, it may be easier to describe in terms of high-level, behavioral equations. As a result, the behavior of these components can be modeled using **Code Modeling**.

Code modeling is the behavioral modeling of devices. A code model consists of a set of interface definitions and a C function implementation describing the component's behavior. The naming and location of these files is important. The model is created by combining two files (`Ifspec.ifs` and `Cfunc.mod`). The resulting file, which is given the same name as the folder containing its source files, is placed in the `codemodl` folder.

5

Simulation

This chapter introduces you to Multisim's simulation functionality. It also discusses Multisim's instruments and analyses.

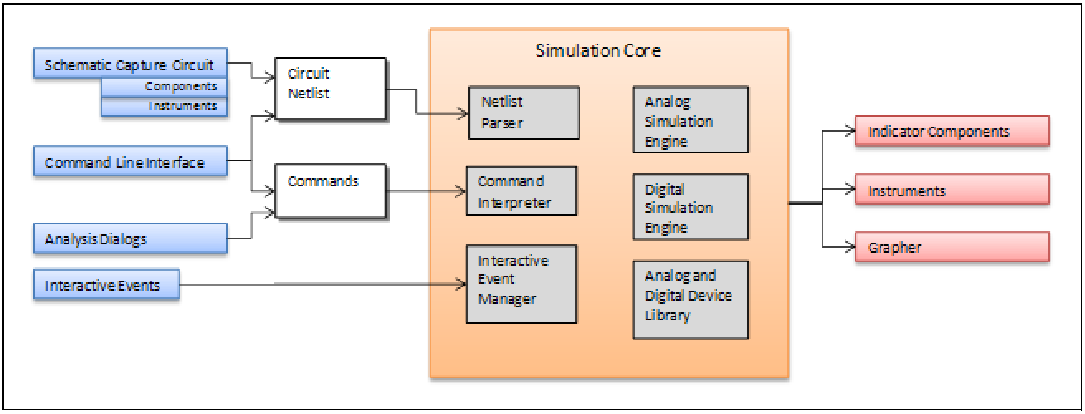
Some of the described features may not be available in your edition of Multisim. Refer to the *NI Circuit Design Suite Release Notes* for a description of the features available in your edition.

Refer to the *Multisim Help* for complete information on the features described in this chapter.

Simulation in Multisim

Simulation is a mathematical way of emulating the behavior of a circuit. With simulation, you can determine much of a circuit's performance prior to physically constructing the circuit or using physical test instruments.

There are several sections in the Multisim software that are closely tied to the simulation process: setting up the circuit to be simulated, setting up the simulation parameters, performing the simulation, and analyzing the results. These sections and their place in the simulation flow are shown in the figure below.



A simulation in Multisim runs in one of two simulation modes: Interactive Simulation Mode or Analysis Simulation Mode.

- **Interactive Simulation Mode**—This mode is used whenever you click the **Run** button. It performs a time-domain simulation (transient analysis) until you click the **Stop** button. In this mode, data is displayed by the instruments and by the indicator components. Additionally, interactive components such as switches and certain instruments such as the **Function Generator**, can be used to change the circuit state during run-time.
- **Analysis Simulation Mode**—This mode is used whenever you press the **Simulate** button in any of the analysis dialogs or whenever you use the **XSPICE Command Line** dialog box to run analyses. This mode allows you to set up a customized analysis and view the results in the Multisim **Grapher** during and after the simulation.

Schematic Capture Circuit
For the purposes of simulation, schematic capture is used to graphically construct the netlist, which is a text-based representation of the circuit, from interconnections of components and instruments. The associated component and instrument models are automatically inserted into the netlist. The netlist is generated and loaded into the simulation core upon invoking any analysis dialog or starting an interactive simulation.

Analysis Dialogs
Analysis dialogs set up analysis-simulation-mode simulation. They automatically generate simulation Commands from the settings within them.

Instruments
Instruments have multiple uses including signal generation and data analysis. Although they are primarily designed for the interactive simulation mode, certain instruments such as the **Function Generator** can be used in the analysis simulation mode as well.

Command Line Interface
The **XSPICE Command Line** dialog box is used as an alternative to schematic capture and analysis dialogs. Use the **XSPICE Command Line** dialog box to load an external, predefined netlist and a set of commands into the simulation core.

Interactive Events
Certain interactive components and instruments can alter the state of the circuit while a simulation is in progress by sending interactive events directly to the simulation core.

Grapher

The **Grapher** is used to analyze the simulation output data. Although it is designed primarily for the analysis simulation mode, certain instruments send their data to the **Grapher** during the interactive simulation mode.

Indicator Components

Indicator components display output values or change their appearance when used in an interactive mode simulation. Indicator components are not responsive in the analysis simulation mode. Commonly-used animated components include the **Ammeter**, the **7-segment display** and the **Probes**.

Simulation Core

The Simulation Core carries out the specified analysis. The core is based on the XSPICE project from Georgia Technical Research Institute. XSPICE adds a native event-driven digital simulator to the SPICE3 analog simulator from the University of California at Berkeley. Both XSPICE and SPICE3 are public domain projects.

The Multisim Simulation Core contains many improvements to the XSPICE core, especially in the areas of convergence and Cadence® PSpice® compatibility. The Multisim Simulation Core also contains an Interactive Event Manager.

Simulation Core Subsystems

The Simulation Core is composed of the following subsystems: the Netlist Parser, the Command Interpreter, the Interactive Event Manager, the Analog and Digital Simulation Engines, and the Analog and Digital Device Library.

The Netlist Parser interprets the circuit netlist and builds an internal representation of the circuit from the devices. Using appropriate syntax, the models in the netlist must reference devices that are in Multisim's Analog and Digital Device library. The Analog Device Library contains many models that closely resemble the behavior of actual electronic elements such as resistors, capacitors, diodes, and MOSFETs. The library also includes more abstract elements, such as voltage sources, controlled sources and ideal switches, which can be used to construct macromodels, such as op-amps. The Digital Device Library includes common elements found in gate-level digital circuits such as NAND gates, NOR gates, and flip-flops. It also includes elements that are customizable with user-defined logic behavior.

The Command Interpreter interprets the commands and sets up the simulation internally.

The Interactive Event Manager waits for events from interactive elements and changes the circuit state's during transient analysis.

The Analog and Digital Simulation Engines perform the actual calculations in accordance with the analysis specified.

Instruments

Multisim includes a number of virtual instruments that you can use to measure the behavior of your circuits. These instruments are set, used and read just like their real-world equivalents. In addition to the standard instruments that come with Multisim, you can create your own custom instruments using LabVIEW, a graphical development environment for creating flexible and scalable test, measurement, and control applications.

Virtual instruments have two views—the instrument icon you attach to your circuit, and the instrument face, where you set the instrument's controls. You can show or hide the face by double-clicking on the instrument's icon. The instrument faces are always drawn on top of the main workspace so that they are not hidden. You can place the instrument faces wherever you wish on your desktop. When you save your circuit, the instrument face locations and show/hide status are stored with the circuit. Any data contained in the instruments is saved, up to a preset maximum size.

Depending on your edition of Multisim, you will have some or all of the following instruments:

- Multimeter
- Function Generator
- Wattmeter
- Oscilloscope
- Bode Plotter
- Word Generator
- Logic Analyzer
- Logic Converter
- Distortion Analyzer
- Spectrum Analyzer
- Network Analyzer
- Measurement Probe
- Four Channel Oscilloscope

- Frequency Counter
- IV Analyzer
- Agilent Simulated Function Generator
- Agilent Simulated Multimeter
- Agilent Simulated Oscilloscope
- Tektronix Simulated Oscilloscope
- Voltmeter
- Ammeter
- Current Probe
- LabVIEW Instruments

Analyses

Multisim offers you many analyses, all of which use simulation to generate the data for the analysis you want to perform. These analyses can range from quite basic to extremely sophisticated, and can often require one analysis to be performed (automatically) as part of another.

When you activate an analysis, the results are displayed on a plot in the Multisim **Grapher**, unless you specify otherwise, and saved for use in the **Postprocessor**. Some results are also written to a viewable audit trail.

Depending on your edition of Multisim, you will have some or all of the following analyses:

- DC Operating Point Analysis
- AC Analysis
- Single Frequency AC Analysis
- Transient Analysis
- Fourier Analysis
- Noise Analysis
- Noise Figure Analysis
- Distortion Analysis
- DC Sweep Analysis
- DC and AC Sensitivity Analyses
- Parameter Sweep Analysis
- Temperature Sweep Analysis

- Transfer Function Analysis
- Worst Case Analysis
- Pole Zero Analysis
- Monte Carlo Analysis
- Trace Width Analysis
- Nested Sweep Analyses
- Batched Analyses
- User Defined Analyses

6

Data Analysis

This chapter introduces you to Multisim's **Grapher**, **Reports**, and **Postprocessor**.

Some of the described features may not be available in your edition of Multisim. Refer to the *NI Circuit Design Suite Release Notes* for a description of the features available in your edition.

Refer to the *Multisim Help* for complete information on the features described in this chapter.

The Grapher

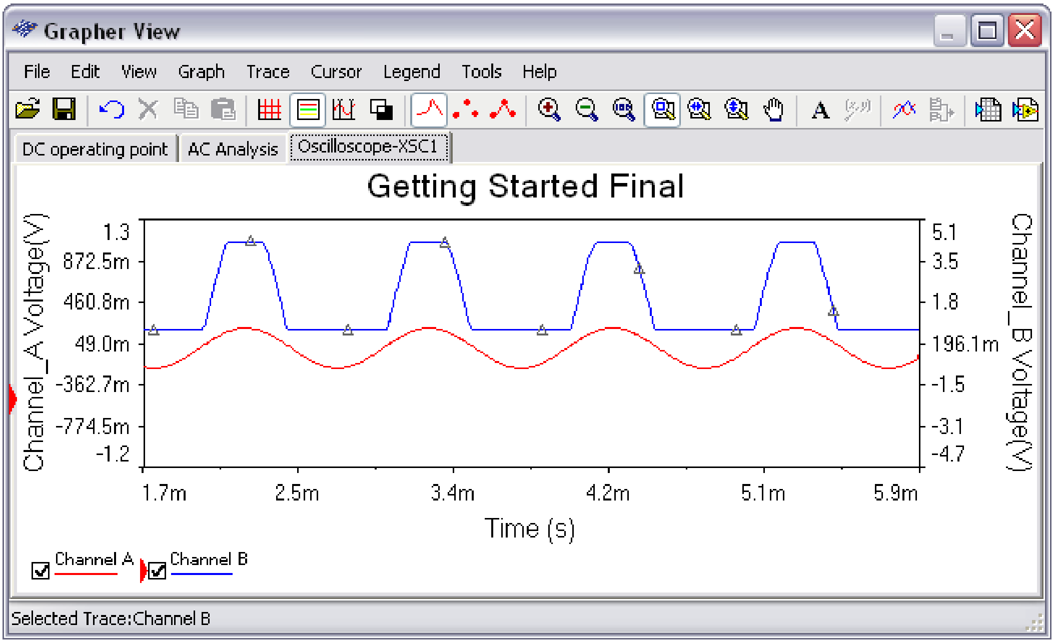
The **Grapher** is a multi-purpose display tool where you can view, adjust, save, and export graphs and tables. It displays:

- the results of all Multisim analyses in graphs and tables.
- a graph of traces for some instruments (for example, the **Oscilloscope** and **Bode Plotter**).

You can also plot extra traces from the results of the most recent simulation in the **Grapher** and export simulation results from the **Grapher** to Excel or LabVIEW.

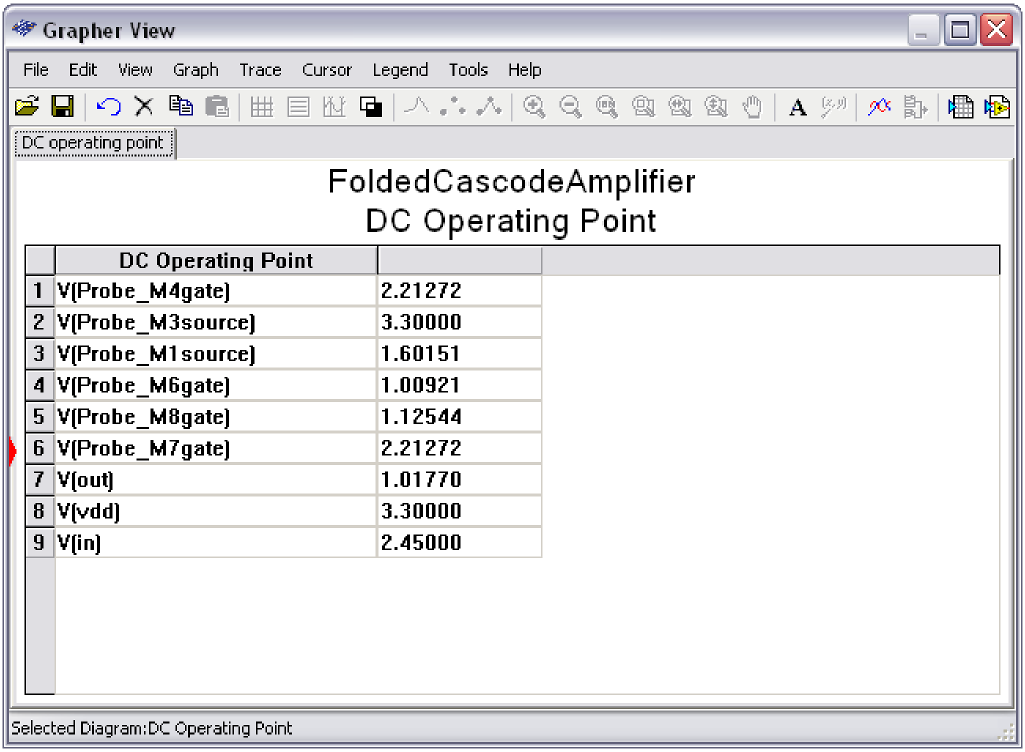
You can save the results of your simulation to either a text-based (`.lvm`) or binary (`.tdm`) measurement file, and use applications like National Instruments LabVIEW and DIAdem to compare simulated output with actual circuit output.

The **Grapher View** shows both graphs and tables. In a graph, data is displayed as one or more traces along vertical and horizontal axes, as shown in the figure below.



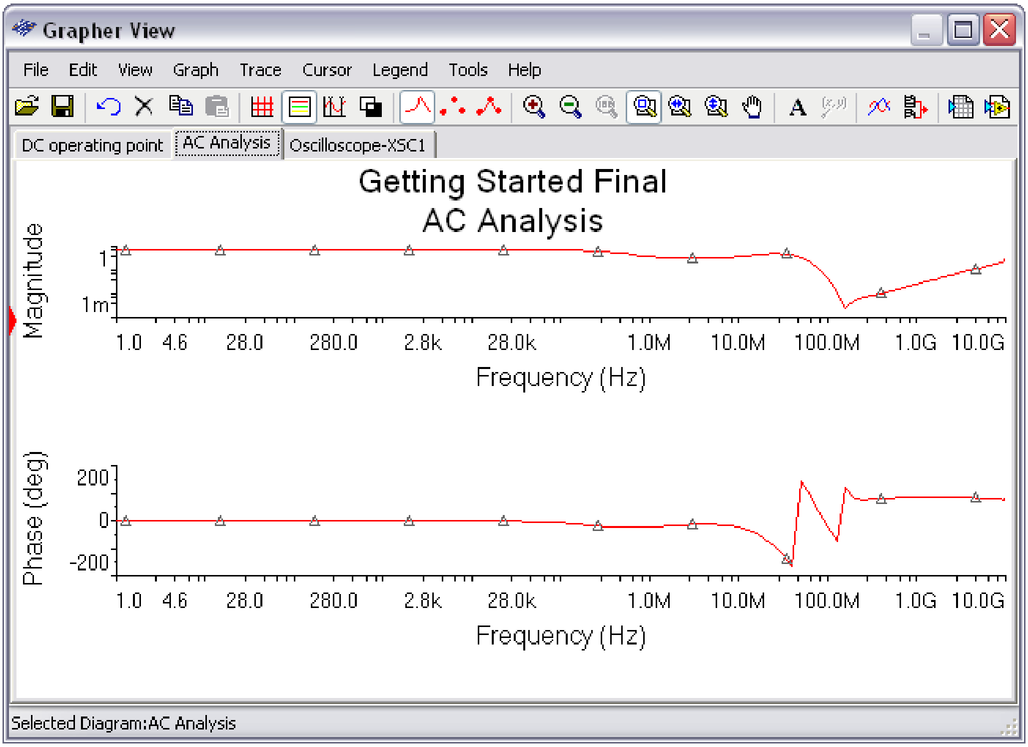
Some analyses can also provide digital graphs. Refer to the *Multisim Help* for information.

In a table, text data is displayed in rows and columns, as shown in the figure below.

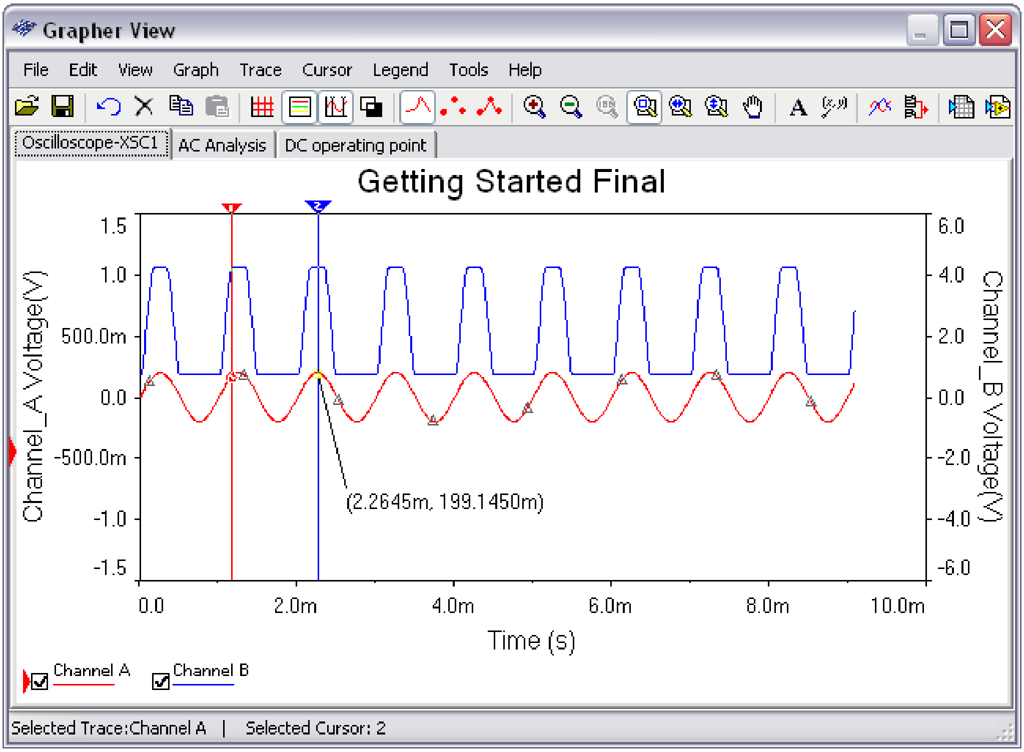


	DC Operating Point	
1	V(Probe_M4gate)	2.21272
2	V(Probe_M3source)	3.30000
3	V(Probe_M1source)	1.60151
4	V(Probe_M6gate)	1.00921
5	V(Probe_M8gate)	1.12544
6	V(Probe_M7gate)	2.21272
7	V(out)	1.01770
8	V(vdd)	3.30000
9	V(in)	2.45000

Some analyses, such as the **AC Analysis**, create both a phase and magnitude graph on the same tab when the analysis is run. In this case, the x-axes (frequency) of the plots line up as shown in the figure below.



You can place a data label, containing the X-axis and Y-axis values, at the point where a cursor crosses the selected trace, as shown in the example below **(2.2645m, 199.1450m)**.



Reports

The following reports are available in Multisim:

- **Bill of Materials**—Lists the components used in a design and provides a summary of the components needed to manufacture the circuit board.
- **Component Detail Report**—Shows all information stored in the Multisim database for a particular component.
- **Netlist Report**—Provides circuit connectivity information for each component in a design.
- **Schematic Statistics Report**—Lists the quantity of components, gates, nets, and other elements of a design.
- **Spare Gates Report**—Lists the unused sections of multi-section components in a design.

- **Model Data Report**—Provides model information for a selected component.
- **Cross Reference Report**—Provides a detailed list of all components in the active circuit, including the sheet they are on.

Postprocessor

You can use the **Postprocessor** to manipulate the output from analyses performed on a circuit and plot the results on a graph or table.

To use the **Postprocessor**, you build equations by combining the variables from previous circuit analysis results with mathematical functions. When you perform an analysis on a circuit, the results appear in the **Grapher** and are stored for use by the **Postprocessor**. To build equations for the **Postprocessor**, you must have performed at least one analysis.

Types of mathematical operations that can be performed on analysis results include: arithmetic, trigonometric, exponential, logarithmic, complex, vector, and logic. Refer to the *Multisim Help* for details.

The following illustrate possible uses of the **Postprocessor**:

- Divide the output curve by the input curve obtained from a transient analysis, and observe the results.
- Multiply a voltage by a current to observe circuit power.
- Assess the differences caused by minor changes to your circuit. For example, run an analysis on a circuit, then change one condition of the circuit (such as changing the input voltage of the component's value) and run the analysis again. Subtract one set of results from the other to show the effect of the circuit modification.

Equations created in the **Postprocessor** are saved with Multisim allowing them to be available in the **Postprocessor** the next time you open Multisim. Your expressions can be used with other analysis results as long as the expression variables are labeled the same.

7

PCB Layout

This chapter discusses transferring of a Multisim design to PCB layout.It also discusses NI Circuit Design Suite's Forward and Backward annotation processes.

Some of the described features may not be available in your edition of Multisim. Refer to the *NI Circuit Design Suite Release Notes* for a description of the features available in your edition.

Refer to the *Multisim Help* for complete information on the features described in this chapter.

Transferring to PCB layout

You typically send a completed Multisim design to Ultiboard for layout.

If you are using a PCB layout package other than Ultiboard, you can create files in the necessary formats for transfer to other layout packages including OrCAD, PADS Layout 2005, P-CAD and Protel.

Forward and Backward Annotation

Forward Annotation is the process of sending schematic design changes to a corresponding PCB layout file.

When you do a forward annotation, the differences between the Multisim file and the Ultiboard file display in the **Forward Annotation** dialog box. You select from a list of possible actions for each difference.

Backward Annotation is the process of sending PCB layout changes to their corresponding schematic design file.

When you do a backward annotation, the differences between the Ultiboard file and the Multisim file display in the **Backward Annotation** dialog box. You select from a list of possible actions for each difference.

Both forward and backward annotations are invoked from within Multisim or Ultiboard and provide the same user experience. A `.ewnet` file is produced during annotation that you can save and share. This file is the synchronization medium for both forward and backward annotation.

8

Automation API

This chapter introduces you to Multisim's Automation API.

Some of the described features may not be available in your edition of Multisim. Refer to the *NI Circuit Design Suite Release Notes* for a description of the features available in your edition.

Refer to the *Multisim Help* for complete information on the features described in this chapter.

Overview

The Application Programming Interface (API) permits the automation of circuit simulation and analyses using Multisim. This is a COM interface that you can use to programmatically control Multisim without directly interacting with it.

The COM interface supports a limited set of circuit editing features. For example, from the client application, you can open a Multisim circuit, simulate the circuit, and then retrieve simulation results for further processing in the client. In addition, the client program can select output variables and substitute client-supplied data for power supplies. Other applications include:

- **Component Evaluator**—This lets you test multiple components under the same circuit topology. You can replace the device under test with others from the database and compare the results. You can also vary the values of the resistors, capacitors, and inductors in the circuit to determine their impact.
- **Stress Analysis**—This is akin to an automated worst-case analysis, where circuit elements are successively replaced by elements that are within the original elements' tolerances to see whether power consumption or other such variables exceed allowed values.

- **Fault Diagnosis**—This simulates a number of faults in a circuit (for example, short circuits, burned-out components, missed connections) that can be compared against a faulty prototype PCB to try to diagnose where the error in the prototype actually lies. This may require the addition of extra circuit elements to simulate the faults.

The COM interface in Multisim can be used by clients written in any COM-aware programming language, such as NI LabVIEW, Visual Basic or C++.

9 Multisim MCU

This chapter introduces you to Multisim's MCU co-simulation functionality.

Some of the described features may not be available in your edition of Multisim. Refer to the *NI Circuit Design Suite Release Notes* for a description of the features available in your edition.

Refer to the *Multisim Help* for complete information on the features described in this chapter.

Introduction

Microcontroller (MCU) components are useful for many circuit designs. A modern microcontroller typically combines a CPU, data memory, program memory, and peripheral devices on a single physical chip. The integration of these essential elements of a computer into a single chip reduces component counts and board size resulting in higher reliability with more capabilities.

You can use the MCU development environment to pause a simulation, inspect the internal memory and registers of the MCU, set code breakpoints and single step through your code.

Note Refer to the *Getting Started with NI Circuit Design Suite* manual for a tutorial covering Multisim's main MCU functions.

Debugging Tools

Multisim's MCU debugging tools give you the ability to control execution at the instruction level while also providing views of the memory and registers within the MCU.

After an MCU's project has been successfully built using the **Build** command, the debug window becomes accessible.

There is one **Debug View** available for each source code file, as well as one for the disassembly of the entire project. By default, opening a **Debug View**

opens the project disassembly view which is a complete disassembly of the ROM for the relevant MCU. In the project disassembly view, every memory address is shown from address zero to the maximum address of the ROM. The project disassembly view always shows the disassembly (not the listing assembly) because it shows memory addresses which may not have a corresponding listing (such as the opcodes from statically linked libraries).

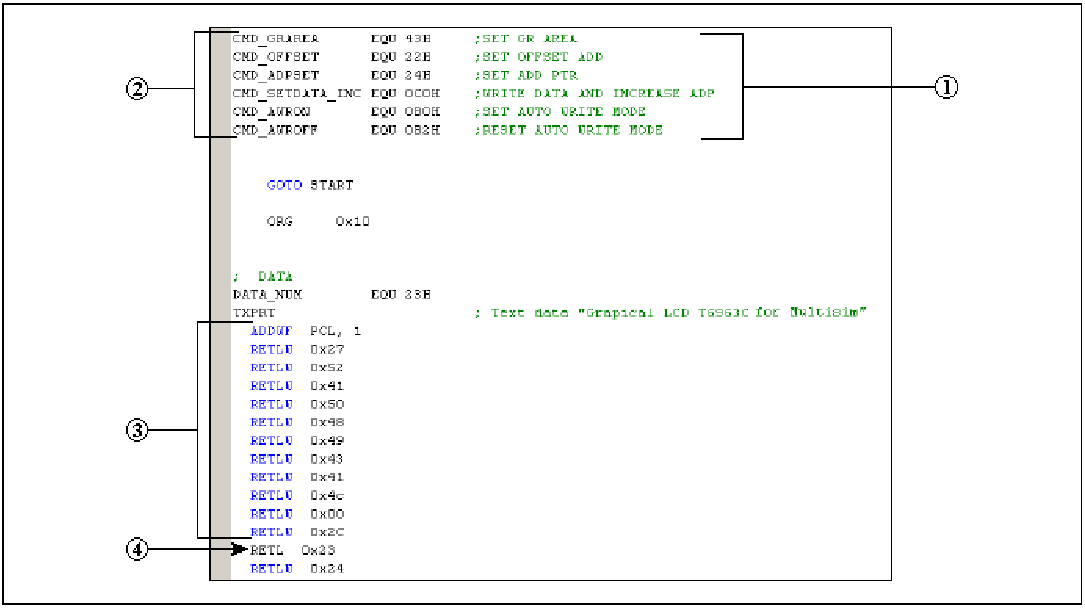
MCU Source Code Editor

The source code view shows the assembly or C source code for the MCU program.

The numbers on the far left are the program memory addresses and the hexadecimal codes to their immediate right are the assembled codes for each mnemonic assembly instruction. The column of numbers in the middle shows the line number in the original assembly or C source. The remainder of the line to the right shows the assembly source and comments.

Sections of the MCU **Source Code Editor** are color-coded to denote the text type, as shown in the figure below.

- `Blue text`—Key words (3).
- `Green text`—Notes (1).
- **`Black text`**—All other text, including data (2), errors (4).



MCU Memory View

The contents of the **MCU Memory View** change depending on the type of MCU. It may, for example, contain internal memory information, register views and configuration information.

Refer to the *Multisim Help* for information about specific MCUs.

Advanced Features

Multisim's MCU functionality includes advanced debugging tools that make it easy to pause your circuit and explore the internal data and state of the MCU controlling your circuit. You can set breakpoints and single step through assembly code while validating that the register contents are changing as expected. You can also manually edit most memory views while debugging.

An example of these debugging features is found in *Getting Started with NI Circuit Design Suite*.

Peripheral Devices

Along with its selection of MCUs, Multisim contains a number of peripheral devices.

The **MCU Group** contains RAM and ROM devices that are designed to function specifically with the MCUs.

The **Advanced Peripherals Group** contains a selection of **Keypads**, **LCDs**, **Terminals** and **Miscellaneous Peripherals** such as the **Liquid Holding Tank**.

Refer to the *Multisim Help* for more information.

MCU Wizard

The **MCU Wizard** helps you get started by creating an MCU Workspace, default project and source file.

Multisim MCU Code Manager

The **MCU Code Manager** dialog box is where you manage the MCU files in an MCU workspace and enter the build settings for each MCU project.

Each MCU project has a number of build settings that consist of assembler/compiler tool information, the command line options to be used with the selected tool, and additional information on the intermediate/output files to generate during a build, and where the tools/files are located.

A

Technical Support and Professional Services

Visit the following sections of the award-winning National Instruments Web site at `ni.com` for technical support and professional services:

- **Support**—Technical support at `ni.com/support` includes the following resources:
  - **Self-Help Technical Resources**—For answers and solutions, visit `ni.com/support` for software drivers and updates, a searchable KnowledgeBase, product manuals, step-by-step troubleshooting wizards, thousands of example programs, tutorials, application notes, instrument drivers, and so on. Registered users also receive access to the NI Discussion Forums at `ni.com/forums`. NI Applications Engineers make sure every question submitted online receives an answer.
  - **Standard Service Program Membership**—This program entitles members to direct access to NI Applications Engineers via phone and email for one-to-one technical support, as well as exclusive access to eLearning training modules at `ni.com/eLearning`. NI offers complementary membership for a full year after purchase, after which you may renew to continue your benefits.

    For information about other technical support options in your area, visit `ni.com/services`, or contact your local office at `ni.com/contact`.
- **Training and Certification**—Visit `ni.com/training` for training and certification program information. You can also register for instructor-led, hands-on courses at locations around the world.
- **System Integration**—If you have time constraints, limited in-house technical resources, or other project challenges, National Instruments Alliance Partner members can help. To learn more, call your local NI office or visit `ni.com/alliance`.

You also can visit the Worldwide Offices section of `ni.com/niglobal` to access the branch office Web sites, which provide up-to-date contact information, support phone numbers, email addresses, and current events.

Index

K

L

M

N

O

P

R

S

T

U

V

W